U.S. ARMY RDECOM

TECHNICAL REPORT RDMR-WD-13-26

DESIGNING AN ELECTRONICS DATA PACKAGE FOR PRINTED CIRCUIT BOARDS (PCBS)

Daniel V. Ryan and **Aaron S. Brown**
Weapons Development and Integration Directorate
Aviation and Missile Research, Development, and Engineering Center

August 2013

AMRDEC

DESTRUCTION NOTICE

FOR CLASSIFIED DOCUMENTS, FOLLOW THE PROCEDURES IN DoD 5200.22-M, INDUSTRIAL SECURITY MANUAL, SECTION II-19 OR DoD 5200.1-R, INFORMATION SECURITY PROGRAM REGULATION, CHAPTER IX. FOR UNCLASSIFIED, LIMITED DOCUMENTS, DESTROY BY ANY METHOD THAT WILL PREVENT DISCLOSURE OF CONTENTS OR RECONSTRUCTION OF THE DOCUMENT.

DISCLAIMER

THE FINDINGS IN THIS REPORT ARE NOT TO BE CONSTRUED AS AN OFFICIAL DEPARTMENT OF THE ARMY POSITION UNLESS SO DESIGNATED BY OTHER AUTHORIZED DOCUMENTS.

TRADE NAMES

USE OF TRADE NAMES OR MANUFACTURERS IN THIS REPORT DOES NOT CONSTITUTE AN OFFICIAL ENDORSEMENT OR APPROVAL OF THE USE OF SUCH COMMERCIAL HARDWARE OR SOFTWARE.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 074-0188

1. AGENCY USE ONLY	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	August 2013	Final

4. TITLE AND SUBTITLE
Designing an Electronics Data Package for Printed Circuit Boards (PCBs)

5. FUNDING NUMBERS

6. **AUTHOR**(S)

Daniel V. Ryan and Aaron S. Brown

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Commander, U.S. Army Research, Development, and Engineering Command
ATTN: RDMR-WDG-C
Redstone Arsenal, AL 35898-5000

8. PERFORMING ORGANIZATION REPORT NUMBER

TR-RDMR-WD-13-26

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSORING / MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

A

13. ABSTRACT ***(Maximum 200 Words)***

The purpose of this report is to define the guidance for designing a Printed Circuit Board (PCB) electronics technical data package. The technical data package is comprised of documentation including an electrical schematic; assembly drawings; Bill of Materials (BOM); PCB data files; component data sheets; packaging specifications; and PCB guidelines for layout, fabrication, and assembly.

14. SUBJECT TERMS
Schematics, Component Data Sheet, Assembly Drawings, Bill of Materials (BOM), Printed Circuit Board (PCB)

15. NUMBER OF PAGES
16

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	SAR

NSN 7540-**01**-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102

TABLE OF CONTENTS

I. BACKGROUND AND OBJECTIVE

The purpose of this report is to provide guidance for program managers and team leaders who are not familiar with the smaller details of acquiring Printed Circuit Boards (PCBs) for the layout, fabrication, and assembly of PCBs for a wide variety of unique applications in a laboratory environment.

The objective of this report is to define and provide guidance for designing a PCB electronics technical data package to help avoid errors and rework.

II. PCB DATA PACKAGE REQUIREMENTS LIST

The technical data package is comprised of documentation including an electrical schematic; assembly drawings; Bill of Materials (BOM); PCB data files; component data sheets; packaging specifications; and PCB guidelines for layout, fabrication, and assembly.

A. Schematics

Schematics should identify each component by its reference designator as it correlates to the assembly drawing and BOM. The value, wattage, voltage, and part number for each component should be included to avoid discrepancies.

B. Component Data Sheet

Data sheets should be provided for each component on the schematics to avoid any discrepancies between vendors. The information on the data sheets should include the vendor, manufacturer, and part number.

C. Assembly Drawings

Assembly drawings should include output assembly drawings and board details, top and bottom side component placement, and specified orientation of electronic and mechanical parts including special instructions, if required. Each part on the drawing must coincide with the BOM.

D. Bill of Materials

Each component in the assembly drawings should have a reference designator; vendor part number; manufacturer part number; component description; quantity; and include special component installation instructions, if required.

Each item on the BOM should correspond with the assembly drawing via reference designators with item numbers indicating the quantity, part number, and description. The BOM should indicate any required specification documents that apply to that assembly level, including test specifications, schematics, firmware, special process specifications, solder mask, and so forth.

E. PCB Data Files

The following list of schematics, drawings, and documents ensure that the PCBs are made to the correct specifications. All of the PCB designs should be created using electronic schematic design Computer-Aided Design (CAD) software such as OrCAD, Altium Designer, or similar design software. If these files were to be given to another manufacturer, the PCBs could be manufactured identically. Pictures, a Portable Document File (PDF), and hand-drawn schematics of PCBs are not acceptable because they cannot be edited or trusted for the exact dimensions and component location.

- Gerber Files
- Aperture List
- Numerically Controlled Drill File
- Drill Report (Drill Sizes for Setup of Drill Operation)
- Fabrication Drawings
- Netlist
- Readme File (Referenced to the PCB with a Description of Each File)
- Assembly Drawings
- Schematic
- Parts Placement

F. Packaging Specifications

The Government Point of Contact (POC) and manufacturer should coordinate the delivery of the assembled products with defined packaging requirements, including boxes, packing materials, labeling, PCB data files, and special handling instructions.

G. PCB Guidelines (Layout, Routing, and Manufacturing)

All of the suggested PCB guidelines can be tailored to adapt to a specific application required by the Government. They should define the layout, routing, and manufacturing of PCB types which include analog, high-speed digital, or a combination of both. For best results, the PCB should be fabricated to Association Connecting Electronic Industries (IPC)-6012 Class 3 guidelines, and the workmanship should conform to an IPC-A-600 Class 3 unless that level of quality is not required. The manufacturer should follow good commercial practices, which include the application of current revisions.

- A suggested layer stack-up should be determined by the Government POC and PCB manufacturer after the initial design review.
- The PCB should be fabricated using Flame Retardant (FR)-4 materials with a glass transition temperature of 170 °C or greater.
- Routing layers should be fabricated with 1.0 ounce per square foot copper.

- Planes should be fabricated with 2 ounces per square foot copper.
- The recommended trace routing is 0.005-inch conductor widths and 0.005-inch spacing.
- The length of parallel trace runs at 0.005-inch spacing on the same routing layer should be limited to 2.0 inches for three or more adjacent traces.
- The two sides of the PCB are referred to as the top and bottom sides. This nomenclature should be used for all documentation used in the layout, design, and fabrication process.
- Differential signal pairs should be routed directly adjacent to each other, depending on PCB complexity. The intent is to reduce differences in trace length and control impedance across the pair.
- No inner layer conductor (trace or plane) should pass within 0.100 inches of any edge of the PCB.
- No conductor (trace or plane) should pass within 0.100 inches of any PCB mounting hole. The minimum space between conductors is determined by the peak voltage difference.
- Clock signals should be routed first to allow for a direct path from point to point without changing routing layers; that is, the entire trace length for each of these signals should be contained within a single routing layer. These clock signals should be isolated from each other and other signals to the extent possible. Routing for these signals should not use either of the external (surface) routing layers.
- All vias and thru-holes used to connect to a ground should make connections to all ground planes in the PCB.
- Pin 1 is the positive (+) terminal of polarized capacitors.
- Reference designators, component outlines, labeling text, and the serial number should be legible on a fully populated PCB, if possible. Serial numbers should begin at “001” and proceed sequentially.
- All nets should be accessible for electrical probing. This should be accomplished through the use of non-solder-masked vias and/or test pads.
- All pads for Ball Grid Array (BGA) solder-bumps must be non-solder-masked-defined, and the pad diameter must match the diameter of the BGA package pad.
- A matte green, liquid photoimageable solder mask should be applied over bare copper traces.
- Component pads should be finish plated with immersion gold. The thickness of the gold should be between 2 and 5 micro-inches.
- The recommended finished PCB flatness deviation should be less than 0.010 inches per inch.

- The minimum copper wall thickness of plated-thru holes should be 0.001 inches with a minimum annular ring of 0.002 inches.
- Electric, flying probe, or x-ray testing should be performed on all bare finished boards (if required), depending on their complexity.

III. PCB COMPLEXITY

The PCB complexity is determined by the circuitry (smallest trace width and clearance space), number of layers, number of holes, and manufacturability of the board. Layout density versus the number of layers used is the primary driver for complexity and schedule. Adding extra layers is also a good solution for solving electrical complexity, although it raises the cost.

IV. IPC-6011 GENERIC PERFORMANCE SPECIFICATION FOR PCBS

Typically, current IPC standards for PCB fabrication and assembly are used by the manufacturer. For the best results, Class 3 High Reliability Electronics for Military use are recommended. If a Class 2 board is required, it should be manufactured using Class 3 requirements for better reliability unless otherwise specified. The following is a list of the different types of PCBs that are used in various applications in electronic equipment:

- IPC-6012 Rigid (Typical PCBs)
- IPC-6013 Flex (Cable)
- IPC-6014 PCMCIA (Personal Computer Memory Card International Association)
- IPC-6015 MCM-L (Multi-Chip Module – Laminated)
- IPC-6016 HDI (High Density Interconnect)
- IPC-6018 Microwave

V. CLASSES OF PCBS

A. Class 1: General Electronic Products

Class 1 products are defined as inexpensive consumer product; computers; and accessories; and lower grade Military hardware, where continued performance and extended life are not required and uninterrupted service is not critical. Cosmetic imperfections are acceptable.

B. Class 2: Dedicated Service Electronic Products

Class 2 products include communications equipment; sophisticated computers; business machines; precision instruments; and Military equipment, where continued performance and extended life are required. Uninterrupted service is preferred, but it is not critical. Certain cosmetic imperfections are allowed.

C. Class 3: High Reliability Electronic Products

Class 3 products include equipment used for commercial and Military products that require continued high performance or critical performance on demand. Equipment downtime cannot be tolerated and the end-use environment may be uncommonly harsh. The equipment must function when required in life support, flight control, and other critical systems. Uninterrupted service is critical, and cosmetic imperfections are not allowed.

VI. DELIVERABLES

The board manufacturer should deliver the following:

- Schedule weekly progress reports to document short turnaround technical issues and their impact on schedule. Otherwise, if no problems arise, a monthly report on individual tasks should be sufficient, depending on the PCB complexity.
- The PCB assemblies should be delivered as scheduled.
- There should be low level test results for electrical, flying probe, or X-Ray testing, if required.
- The PCB data files should be in an electronic CAD format.

VII. CONCLUSION

If these guidelines are followed, most of the risk should be mitigated during the design, fabrication, and assembly of PCBs.

LIST OF ACRONYMS, ABBREVIATIONS, AND SYMBOLS

°	degree
BGA	Ball Grid Array
BOM	Bill of Materials
C	Celsius
CAD	Computer-Aided Design
FR	Flame Retardant
GFN	Glass-Filled Nylon
HDI	High Density Interconnect
ICT	In Circuit Tester
IPC	Association Connecting Electronics Industries
MCM-L	Multi-Chip Module – Laminated
MIL	Military
NEMA	National Electrical Manufacturers Association
PCB	Printed Circuit Board
PCMCIA	Personal Computer Memory Card International Association
PDF	Portable Document File
POC	Point of Contact
UL	Underwriters Laboratories
SMOBC	Solder Mask Over Bare Copper
SMT	Surface Mount Technology
STD	Standard

APPENDIX A:
DEFINITIONS

The definitions for the assembly drawing are as follows:

- **Annular Ring**—A designated pad with a hole (via) that connects a trace on one layer to a trace on another layer.

- **Annular Ring Width—**(diameter of a designated pad – diameter of the hole)/2.

- **Aperture List**—A text file describing the size and shape of each element on the board, also known as a D-code list. This report is not necessary if the files are saved as Extended Gerber with embedded Apertures (RS274X).

- **Aspect Ratio**—The Printed Circuit Board (PCB) thickness divided by the diameter size of a drilled hole.

- **Base Material—**The Flame Retardant (FR) base material is specified along with any specifications that must be met. The FR-4 glass fiber/epoxy resin is a National Electrical Manufacturers Association (NEMA) grade laminate that is a standard base material for PCB's. The Glass-Filled Nylon (GFN) per Military (MIL) Standard (STD)-13949/4D is equivalent to a specific grade of NEMA FR-4.

- **Board Markings—**The requirements and board location for the manufacturers' trademark, date code, and Underwriters Laboratories (UL) marking.

- **Copper Thickness (Weight)—**The weight of copper per square foot that is needed for the inner and outer layers (1 ounce is typical); specify finished copper weight.

- **Electrical Testing—**A fixture that uses a “bed of nails” for testing circuit continuity.

- **Flying Probe Testing**—An In Circuit Tester (ICT) that uses electro-mechanically controlled probes to access nodes on a PCB including boundary scan testing, if required.

- **Layers—**The number of conductor layers that applies to the design (that is, single sided, two layer, four layer, and so forth). For multi-layer boards, the layer stackup should be detailed.

- **Performance Class**—Determined by the end use of the product, which will also determine how the PCB supplier manufactures the PCB. In general, the higher the class, the more expensive and complex the PCB. (Refer to PCB Classes.)

- **Schematic—**A drawing that explains the connectivity between components in an electronic circuit.

- **Silkscreen—**The color (white is typical) of the silkscreen and the degree to which silkscreen is allowed to be removed to avoid pads and plated through holes.

- **Solder Coating—**Solder Mask Over Bare Copper (SMOBC) refers to the finished product where the PCB should have the solder mask coating applied to the bare copper and then the PCB should have a tin-lead coating applied to the remaining exposed copper areas.

- **Surface Mount—**Leads (pins) on the chips and components are soldered on top of the PCB. Fabrication is faster because Surface Mount Technology (SMT) eliminates drilling thru-holes.

- **Thickness—**The finished thickness and tolerance of the base material. (The .062 inches +/- .007 inches or 1.57 millimeter are typical.)

- **Thru-hole**—Leads (pins) on discrete components and chips are inserted through holes in the PCB and soldered from the bottom.

- **Via—**An electrical connection on a PCB that can carry a signal or power between layers.

- **Via Aspect Ratio—**The ratio of the PCB thickness to the smallest unplated drilled hole diameter.

- **X-Ray Testing—**Nondestructive inspection to identify PCB and solder imperfections including opens and shorts.

APPENDIX B:
ASSEMBLY DRAWING DESCRIPTIONS

Assembly drawings should accurately identify the proper orientation of electronic and mechanical parts on the PCB.

- **Adhesives—**Specify adhesives as generically as possible by using terms such as "or equivalent." Specify acceptability criteria/diagrams when the application of the adhesive is critical or when the application of the adhesive cannot be relinquished to general workmanship standards.

- **Component Height—**Specify the maximum and minimum heights where appropriate when the height of a component may fluctuate upon assembly and the height is critical to next higher level assemblies.

- **Component Perpendicularity—**Specify the perpendicularity of components when the control of the element cannot be relinquished to general workmanship standards.

- **Conformal Coating/Map—**Identify areas of conformal coat coverage. Specify dimensions of coverage acceptability when critical.

- **Lead Protrusion Height—**Specify lead protrusion minimum and maximum heights when the control of the heights cannot be relinquished to general workmanship standards.

- **Marking/Labeling—**Specify marking/labeling requirements as generic as possible to allow for manufacturing flexibility. Identify positioning and content to the minimum requirements.

- **Order Sequence of Multi-Level Parts (Hardware)—**Specify the stacking order/position of hardware components.

- **Switch Settings—**Specify switch settings when specific product switch settings are required upon shipment.

- **Torques—**Specify minimum and maximum torque requirements when the control of the torques cannot be relinquished to general workmanship standards.

- **Unfilled Plated Holes—**Specify the plated holes that are to be free of solder for use at the next higher level assemblies or for use by the Government.

- **Wire/Cable Positioning—**Specify the exact positioning, routing, and length of wires or cables when critical.

INITIAL DISTRIBUTION LIST

		Copies
Weapon Systems Technology Information Analysis Center Alion Science and Technology 201 Mill Street Rome, NY 13440	Ms. Gina Nash gnash@alionscience.com	Electronic
Defense Technical Information Center 8725 John J. Kingman Rd., Suite 0944 Fort Belvoir, VA 22060-6218	Mr. Jack L. Rike jrike@dtic.mil	Electronic
AMSAM-L	Ms. Anne C. Lanteigne hay.k.lanteigne.civ@mail.mil	Electronic
	Mr. Michael K. Gray michael.k.gray7.civ@mail.mil	Electronic
RDMR		Electronic
RDMR-CSI		Electronic
RDMR-WDG-C	Mr. Ken W. Pruitt kenneth.w.pruitt2.civ@mail.mil	Electronic
	Mr. Patton Bradford glenn.p.bradford.civ@mail.mil	Electronic
	Mr. Aaron S. Brown aaron.s.brown.civ@mail.mil	Electronic
	Daniel V. Ryan daniel.v.ryan.civ@mail.mil	Electronic/Hardcopy